

STATUS OF COMPUTATIONAL FLUID DYNAMICS IN THE UNITED STATES

Paul Kutler,* Joseph L. Steger,[†] and F. R. Bailey[‡]
NASA Ames Research Center, Moffett Field, California

Abstract

Computational fluid dynamics (CFD) is beginning to play a major role in the aircraft industry of the United States because of the realization that CFD can be a new and effective design tool and thus could provide a company with a competitive advantage. It is also playing a significant role in research institutions, both governmental and academic, as a tool for researching new fluid physics, as well as supplementing and complementing experimental testing. In this paper, some of the progress made to date in CFD in the United States is reviewed. Although CFD has applications in many other disciplines, in this paper the discussion is limited to aerospace applications. The paper addresses the status of CFD in the United States in terms of methods, examples of CFD solutions, and computer technology. In addition, the role CFD will play in supporting the revolutionary goals set forth by the Aeronautical Policy Review Committee established by the Office of Science and Technology Policy is noted. The need for validated CFD tools is also briefly discussed.

Introduction

The long-range goal of computational fluid dynamics (CFD) is to develop computer programs that in a reasonable amount of time (say a maximum of 15 min) can compute the viscous flow around realistic aerospace vehicles and their components and reproduce the flow field generated by that same vehicle in actual flight. This capability should produce solutions that depict the detailed fluid physics, as well as predict the vehicle's stability and control, performance, and thermal and aerodynamic loads in a time frame commensurate with design applications.

Steps such as those outlined in the National Aeronautical R&D Goals¹ should provide many challenges for the fluid dynamicists who use wind tunnels, computers, and flight-test vehicles as tools of their trade. They include the following.

1) Subsonics goal: To build trans-century renewal that envisions technology for an entirely new generation of fuel-efficient, affordable transport aircraft

2) Supersonics goal: To attain long-distance efficiency by developing pacing technologies for sustained supersonic cruise capabilities

3) Transatmospherics goal: To secure future options by pursuing research toward the goal of routinely cruising and maneuvering into and out of the atmosphere with takeoff and landing from conventional runways

Pioneering and demonstrating new technologies within the discipline of CFD will help in pursuing the ambitious goals outlined above.

Commercial aircraft during cruise generate flow fields that are primarily attached and mainly inviscidly dominated, whereas fighter aircraft during maneuvers produce flows that are viscously dominated and contain large regions of embedded separated flow. Computer-generated solutions about these configurations are currently being attempted and have been successful in simulating the global features of the flows about these types of aircraft. However, the details of the fluid physics, such as the unsteady flow behavior, are missing and awaiting refinements in the CFD solution procedures. For components of aerospace vehicles such as inlets, nozzles, wings, and ducts, CFD can predict the viscous flow. The accuracy of these results is limited by discretization error, convergence and order of accuracy of the numerical algorithm, assumptions placed on the governing equations, modeling such as the turbulence and gas properties, and, to a certain extent, the computer resources available. Because of these approximations and, in some cases, uncertainty in describing the fluid physics, the computer codes must be validated to build user confidence.

As mentioned above, CFD is experiencing greater visibility by the aerospace community as a tool to aid in the aerospace vehicle design process. Along with its acceptance comes the requirement by the users for validation; that is, a measure of the accuracy of the results produced by the computer code and its range of validity. Because of this understandable and justifiable requirement, CFD is beginning to play a dominant role in stimulating experiments and developing advanced instrumentation.

The aerospace research community is undergoing what can be viewed as a cultural change. Often in the past, computationalists and experimentalists worked somewhat autonomously. The experimenters performed their experiments to understand the fluid physics or obtain design information and compared their data with available theory, and the computationalists performed their

*Chief, Fluid Dynamics Division. Fellow AIAA.

[†]Senior Staff Scientist, Fluid Dynamics Division. Associate Fellow AIAA.

[‡]Chief, NAS Systems Division. Member AIAA.

This paper is declared a work of the U.S. Government and therefore is in the public domain.

calculations and compared their results with available experimental data, theory, or other numerical results. This process involved little or no communication between the two camps. It has been said that "No one believes the analysis except the engineer who performed the calculations. Everyone believes the data except the engineer who performed the tests."

Because the computationalist can now compute some flow features that the experimentalist cannot easily measure or visualize, the two camps are beginning to work more closely together. As a result, better experimental data are being extracted from the experiments to validate the computer codes. Both parties are now contributing as equal partners in this evolutionary process. This cultural change, as any, takes time, but it is happening, and the results should be enhanced CFD design tools.

In the following sections some of the latest algorithms and solution methodology required for efficient CFD solutions will be discussed. In addition, examples of CFD solutions for both internal and external, steady and unsteady flows are presented that demonstrate the current capability in the United States. Finally, CFD enabling computer technologies including supercomputing work stations, networks, and remote access are discussed.

Methods

Because CFD can treat the nonlinear phenomenon that is inherent in high-Reynolds-number flow simulation, it is becoming an integral part of the study of fluid mechanics and the aerodynamic design process. In some ways CFD tends to be a mature science that has demonstrated engineering utility, but the difficulty of the nonlinear flow phenomenon and the complexity of aircraft configurations are such that in many ways CFD remains in its early infancy. As a result, a diversity of computational algorithms is still being developed and refined to treat myriad problem areas in CFD. For the most part, CFD research has spanned the disciplines of fluid mechanics, numerical analysis, and computer science. Research topics in CFD are generally directed toward solving problems for understanding flow physics, treating realistic configurations, maintaining good computational efficiency and stability, postprocessing of the computed results, and maintaining user friendliness. All of these concepts are interrelated, although they are often researched independently. In this section we will briefly review the status and future prospects of some of this CFD methodology and make some personal assessments.

Physical Modeling and Resolution

In most engineering aerodynamic applications, continuum and perfect-gas assumptions apply, and

it is generally assumed that the Navier-Stokes equations adequately describe the flow. However, the flow Reynolds number is generally too high to numerically resolve all of the features (length scales) governed by the Navier-Stokes equations and so various approximate equations are used to model the fluid physics. CFD has made considerable progress in simulating inviscid flow physics. Indeed the methods developed for inviscid transonic and supersonic flow are somewhat responsible for making nonlinear CFD methods a useful engineering tool. Likewise the physics of attached laminar and some turbulent viscous shear layers can also be correctly modeled using boundary-layer theory, parabolized Navier-Stokes schemes, or Reynolds-averaged Navier-Stokes schemes. But considerable work remains in describing and understanding three-dimensional unsteady and vortical flows, thin-shear-layer separation, transition, and turbulence of high-Reynolds-number flow.

The flow over a simple symmetric streamlined body at high angle of attack, for example, can yield possible asymmetric vortex shedding and unsteadiness. The computational modeling of such flows using Reynolds-averaged Navier-Stokes equations--for example, as described in Refs. 2-10--is proving to be as difficult and fascinating as the wind-tunnel and flight experiments. In many ways, correct simulation is a resolution and, therefore, an efficiency problem; the unsteady Navier-Stokes equations are assumed to handle the physics properly if the flow details are resolved. The fact that this detail cannot adequately be resolved with current numerical algorithms and computers necessitates modeling of transition and turbulence phenomena. It is thus widely acknowledged that "Turbulence modeling is becoming a pacing item in the development of CFD codes for many engineering design applications" (Ref. 11, p. 78).

In the United States, work in the study of turbulence is mainly being carried out at NASA laboratories and at several universities. Much of the effort is either being directed toward developing turbulence models for use in Reynolds-averaged Navier-Stokes (RANS) equations or is concerned with direct and large-eddy simulation.¹²⁻¹⁵ In direct simulations, all of the scales of turbulence are calculated, thus requiring a very fine grid, whereas in large-eddy simulations, only the small scales of turbulence are modeled, and the large scales are computed. Simulations involving this approach have been limited to very simple geometries and low-Reynolds-number flows. As shown by some of the latter results, quite interesting flow phenomena are now being computed for small regions of ideally constrained flow by three-dimensional direct-simulation codes. However, these codes have not progressed past microscopic segments of the flow. Nor have current turbulence models been fully tested and refined in three-dimensional RANS simulations of flow about wings or bodies using adequate grid resolution. So how good the models are and how

small the small-scale effects are that have to be resolved remain to be determined. What is becoming clear, however, is that as both computer technology and numerical procedures improve, the two approaches for treating turbulence will begin to merge. The RANS codes will use ever-refined grids and begin to capture more of the larger scales of turbulence, and the direct simulation and large-eddy codes will simulate higher-Reynolds-number flows for more complicated configurations.

There is currently a resurgence of interest in hypersonics, brought on to a large extent by a requirement for an advanced space transportation system.¹ Consequently work is once again under way to include real-gas effects into the flow-simulation codes.^{16,17} Real-gas simulation introduces additional and often very stiff species rate equations that must be solved with the Navier-Stokes equations. Moreover, to accurately solve the species equations, chemical reaction-rate constants must be determined either experimentally or computationally. Unlike perfect-gas continuum flow in which the Navier-Stokes equations can provide the flow physics, all of the fluid physics is not always understood for chemically reacting flow and rarefied gas flow.¹¹

Discretizational Aspects

As noted in the external aerodynamics portion of the NRC study, Current Capabilities and Future Directions in Computational Fluid Dynamics, "The most pressing need is for algorithm development that will lead to practical methods for solving the nonlinear flow equations--full potential, Euler, and the various forms of Navier-Stokes equations--about arbitrary configurations." (Ref. 11, p. 32). The first step in developing such algorithms is to select a method of discretization.

Discretization of the nonlinear flow-field equations using finite-difference, finite-volume, or finite-element schemes can be carried out using either structured or unstructured meshes. Typically, finite-difference and finite-volume methods use structured grids, and finite-element schemes are formulated for unstructured meshes, although, in fact, any of the schemes can use structured or unstructured grids.

In the past, most CFD applications have been carried out using body-conforming curvilinear structured grids, and relatively simple geometric configurations have been treated. This has been appropriate for the class of computing machines previously available and for gaining experience in solving the nonlinear equations.

Body-conforming curvilinear meshes are generally used in finite-difference and finite-volume computations for a variety of reasons: 1) to simplify the application of boundary conditions, 2) to allow clustering of grid points in flow-field action regions, and 3) to help maintain the well-orderliness that is useful for vector

processing and for various implicit methods that employ approximate-factorization techniques such as alternating-direction implicit (ADI). Moreover, the calculation of high-Reynolds-number viscous flow is simply impractical without the use of a body-conforming curvilinear mesh that employs clustering in the direction nearly normal to the body surface.

Generally a single body-conforming curvilinear grid becomes impractical for very complex geometry configurations, because grid lines become too skewed or poorly clustered. Currently, most research in treating complex configurations is being directed toward either using a composite of structured grids or using unstructured grids. Some examples of three-dimensional composite grid codes in the United States are described in Refs. 18-25. Examples from some of these procedures are detailed in the results section. Unstructured grid codes are also finding their way into CFD applications, an example being that of Ref. 26. Both approaches require a more complex data-handling program than that required by a simple single-structured grid. The best approach is still not resolved, although a grid that is structured in most areas and only unstructured in limited regions may ultimately prove to be best for treating complex configurations.

Euler and Navier-Stokes General-Purpose Codes

In the United States a major CFD effort has been directed toward the development of general-purpose flow codes that solve either Euler or Reynolds-averaged Navier-Stokes equations. Simulation codes that treat the Euler equations have tended to replace potential codes for solving nonlinear inviscid flow, although the potential schemes are still widely used and further developments are being pursued as discussed later. The three-dimensional flow-simulation codes used by the aerospace community in the United States are currently based on either finite-volume or finite-difference formulations on structured grids, although as noted previously, the use of unstructured grids is increasing. Considerable research continues in the use of finite-element, weighted-residual, and spectral methods, but these methods have not become an essential part of a general-purpose code. Of the finite-volume and finite-difference codes, there is a choice between either upwind or central-difference schemes. At this point, the more established codes use central differencing,^{8,10,27-30} and the newer codes tend to use upwind^{4,31-35} or a combination of upwind and central differencing.⁶ In either case, there is considerable activity in implementing total variational diminishing (TVD) and flux-difference schemes to better capture strong shock waves (e.g., Refs. 32, 36, 37).

Considerable work is under way to speed up the general-purpose codes and to make them more efficient and more accurate. Enhanced steady-state convergence is being achieved by learning

how to apply classic relaxation techniques to the Euler and Navier-Stokes equations and include the use of multigrid schemes,^{38,39} alternate sweep or symmetric SOR,^{31,33,35} governing equation conditioning,⁴⁰ space varying relaxation or Δt conditioning, acceleration or extrapolation to a steady state,^{41,42} and so on. For predominately supersonic and hypersonic steady flow, codes that use marching techniques where appropriate have been developed.^{43,44}

Various efforts to reduce the computational work per point are under way. For viscous flows, wall functions have been introduced to avoid having to resolve the Navier-Stokes equations on an extremely fine grid near the wall.⁴⁵ Another approach to accomplishing the same objective, termed fortified Navier-Stokes,⁴⁶ allows a fine-grid boundary-layer scheme to be embedded into the Navier-Stokes equations in the very-near vicinity of the wall. There has also been work to reduce the cost per time-step in implicit codes by using simplified matrices that are similar to the original linearization matrices.⁴⁷⁻⁴⁹

Demonstrated improvements in solution accuracy have resulted from using higher-order discretizations, perturbation about approximate solutions, adaptive grids, and schemes that use approximate Riemann solvers. These techniques are being integrated into general-purpose simulation codes.

Zonal Equation Methods

An apparent way to economize on the solution of a flow field is to use a zonal-equation approach. In this technique, simplified equation sets are used in regions of the flow where appropriate. For a uniform incoming flow, for example, a zonal code might use boundary-layer equations near the wall, Euler equations away from the wall, Navier-Stokes equations in separation regions, and isentropic equations in the far field. Various efforts have been made in this direction in the United States, but no general-purpose three-dimensional code has emerged that utilizes this approach. The advantages of the zonal method are that simplified equations can be used (saving computer work), and these equations are often better conditioned to the zone of interest (leading to better numerical convergence). The drawbacks to the zonal method are that the coding becomes more complex and the different equation sets must be interfaced. Poor interfacing can lead to inconsistencies, degraded efficiency, and even numerical instability. Moreover, inconsistencies in the solution can result if the approximate equations are used past their range of validity--a range not always easy to determine. For example, how fast does the boundary-layer approximation break down as one moves away from the wall, and can these equations be interfaced at this point with the next set of approximate equations?

Because of these problems, there are several studies under way to avoid the usual zonal approach. Instead, one general-purpose code is used throughout and either physical terms are removed or approximate solutions are forced into the general solver to improve convergence or resolution. Some of the potential-like schemes described below can fall into the first category, and the fortified Navier-Stokes approach⁴⁶ is an example of the second category.

Potential-Like Schemes

Stream-function and other potential-like formulations of the governing equation have proved to be well adapted to simulating low-speed inviscid and viscous flow, and nonlinear full-potential schemes have been the mainstay of inviscid transonic flow solutions. Although transonic potential schemes are still extensively used in the United States, they seem to be currently out of vogue, perhaps because for complex flow conditions potential and potential-like schemes are often more difficult to code than primitive-variable methods. Research continues on a reduced level, however, and for a variety of reasons it would be a mistake to dismiss this class of methods. An effort is under way, for example, to combine a linear panel method with nonlinear full-potential equations to simulate transonic flow about complex configurations.^{50,51} In this approach, the linear panel scheme is used to enforce the surface boundary condition on an otherwise rectangular grid.

In addition, various researchers have explored adding the vortical flow terms using stream-function formulations, Clebsch functions, and vector potential functions.⁵²⁻⁵⁶ In the latter cases, these functions need only be added in rotational flow regions or they can be used to eliminate circulation cuts in the field. When using these formulations in three dimensions there have been questions about what are the proper boundary conditions. This now seems to have been sorted out for many practical applications. A more serious problem is that shock waves cannot be captured automatically except under an isentropic formulation. This means that a process of imposing the correct shock-jump relation must be included in the program. For supersonic to subsonic flow shocks, locating the shocks and making the correction is fairly straightforward,^{54,57} but supersonic to supersonic shocks can be difficult to locate and correct, especially in three dimensions.

Computer-Algorithm Synergism

The introduction of faster computers is a driver in the development of much more efficient numerical algorithms. This is because as the machines become more powerful, it is easier to experiment with new ideas for numerical algorithms. New algorithms can be quickly verified, problem areas can be isolated, and optimum convergence parameters can be found. As a result,

improvements in numerical algorithms have kept pace with improvements in computer hardware. For example, during an 8-year period, the simulation code described in Refs. 27 and 28 was run an order of magnitude faster because of increased computer speed. But it was run another order of magnitude faster because the solution algorithm was improved as detailed in Ref. 28. A cursory look at the literature describing other codes--for example, Refs. 4, 29, 35, 44--shows a similar trend.

It is generally felt that much of the future increases in computer speed will be due to increased use of multiprocessing elements, for example, parallel and cluster architectures. Perhaps gains in algorithm efficiency will no longer keep pace with computer speed increases owing to massively replicated processing elements. It is important, however, that the numerical algorithms be kept compatible with these architectures. This requirement has long been recognized, and most explicit and many implicit algorithms that are written for an ordered grid can be readily vectorized. All of the codes referenced immediately above, for example, run in a vectorized mode. Implicit schemes, whose inversion processes are typically recursive, have often been vectorized by performing a disconnected set of recursive operations simultaneously so that a vector length equal to the number of simultaneous operations is achieved.

Postprocessing

Now that three-dimensional viscous flow simulations are indeed occurring, the problem of extracting all of the useful information from the large generated data bases is becoming more acute. As some of the three-dimensional solutions discussed later will make clear, flow phenomena such as flow-reversals, shocks, shear-layers, and vortices can often be difficult to identify and visualize, especially if the flow is also unsteady. As a consequence, much of the computational aerodynamicist's time is now being devoted to extracting, displaying, and analyzing various features of the solution. Graphic displays of contour surfaces, particle paths, and the like can also be expensive to generate. In the future more computer resources may be expended on analyzing a solution and displaying it in a meaningful way than what was needed to generate the solution in the first place.

A means of extracting information from a solution is by use of computer flow visualization often using graphic work stations. An advantage of using such a computer flow visualization is that one can inadvertently come across information that might not otherwise be anticipated. Using a graphics work station, an engineer can display various portions of the flow field, observe the solution from different vantage points, and otherwise interact with the computed solution, for example, by seeding particles and tracing their behavior forward and backward in time. Because

the three-dimensional data base can be quite large, the data reduction process is too time-consuming unless either a very powerful graphic work station is available, or the supercomputer itself is used in an interactive manner. In either case, very-high-speed data links between the graphics station and the supercomputer are needed.

Ultimately the supercomputer should be programmed to analyze the data base. Such a computer program is needed in order to thoroughly search a flow field to find details that a human at a work station would find too tedious to locate. But this means that reliable and efficient algorithms must be developed to search out and display special features such as shock waves, vortices, and separation lines. Some of these problems have been described in Ref. 58.

Another important part of post-processing a solution is error analysis. The accuracy of a finite-difference solution is generally appraised by successively refining the grid and comparing solutions from one grid to another. If the solution is unchanged, the flow result is likely resolved. (It is assumed that a Reynolds-averaged Navier-Stokes solution becomes invariant with grid refinement.) Because the next generation of supercomputers will not be powerful enough to carry this process too far, the problem of determining solution accuracy may stimulate development of simple approximate methods and checks that might be used from a work station. For example, a boundary-layer solution to verify the correctness of the skin friction and heat transfer found from a Navier-Stokes procedure might be computed. By taking edge conditions within the viscous layer from the Navier-Stokes result, the boundary-layer equations can be solved using a very refined grid near the wall. If the boundary-layer solution returns the same wall values computed from the Navier-Stokes equations, then the Navier-Stokes solution is adequately resolved.

Computational Complexity

The chief task of the computational fluid dynamicist has been to develop accurate and numerically stable algorithms to solve the inviscid and viscous nonlinear flow equations about various configurations. Because the equations are nonlinear, and, therefore, not fully subject to analysis, there is considerable art and craftsmanship involved in this process. As a result, the building of flow-simulation codes has often been undertaken by one or a few individuals. As the codes have grown with increased physical modeling (e.g., chemically reacting species, turbulence modeling), geometrical capability (e.g., using composite grids), and improved efficiency (e.g., zonal governing equations, multitasking), they have also grown considerably in complexity. More codes are being developed by teams of researchers, and efforts at using artificial intelligence (AI) techniques are emerging.⁵⁹ For

example, the TNS code^{24,60} was undertaken at Ames Research Center as an experiment in building a general-purpose flow solver for compressible flow applications using a team effort. Team-like efforts, which are often foreign to the existing craftsman mentality of code builders, necessitate the use of structured coding practice and an organized effort. In the future, codes will have to be constructed in modules, in an open architecture, such that various specialized nonlinear flow solvers can be easily added or removed. For the near term, it is also likely that more organizations will establish special groups just to run, rather than to develop, the codes for the designer.

Examples of CFD Solutions

The capability of CFD to effectively and accurately simulate the complexities of fluid flows of practical interest has grown tremendously relative to the capability of even a few years ago, and stands as a testament to the ingenuity and imagination of both the CFD scientists and the computer system developers. Whereas only a few years ago, there were very few simulations utilizing the Navier-Stokes equations or Reynolds-averaged Navier-Stokes equations, today a significant fraction of the research and development effort in CFD involves simulations utilizing these equations. The aircraft industry is now employing Euler CFD programs as part of its design process to supplement their linear panel method analysis tools. Eventually, the Navier-Stokes development work being performed in research laboratories will attain a level of efficiency to also be used routinely by the aerospace industry.

Several examples will demonstrate the strides that have been made in recent years, and help illustrate the state-of-the-art CFD in the United States. In the following discussions, the examples are divided into aerospace applications and fluid physics applications. The aerospace applications section is further divided into internal and external flows.

Aerospace Applications: Internal Flows

An example of the application of supercomputers to the analysis of internal flow problems is the Euler or Navier-Stokes turbomachinery code developed by Chima⁶¹ of the NASA Lewis Research Center. The effects of radius change, stream surface thickness, and rotation are included, allowing calculations for centrifugal impellers, radial diffusers, and axial machinery with contoured end walls (Fig. 1). This code was used in the calculation of flow in a 6:1 pressure ratio centrifugal impeller used for an auxiliary power unit. Figure 2 shows computed relative Mach-number contours. The solution shows a supersonic pocket on the leading edge, terminated by a shock. Rotational effects make the suction-surface boundary-layer thin, the pressure-surface

boundary-layer thick, and causes the wake to leave the trailing edge in a spiral.

To model the viscous flow through rotor-stator systems, it is necessary to simulate the flow about multiple moving bodies. Computationally modeling such a flow usually involves one grid system moving relative to another, with the attendant requirement that information be passed across the interface boundary in a time-accurate basis. Rai at Ames Research Center developed such a solution procedure that has resulted in the ability to simulate the flow through such systems in both two and three dimensions.^{22,62} The results have been spectacular in terms of visual and flow-field detail. The flow fields for both supersonic and subsonic blade rows have been simulated. Using present graphic work-station technology, it is practical to show the development of alternating trailing-edge vortices in the subsonic blunt-airfoil case, and how these are propagated through the cascade system. The extension to three dimensions has recently been accomplished, with results obtained for a turbine rotor-stator system, including the hub and tip effects. Velocity vectors for such a system are presented in Fig. 3. Such a simulation capability gives the turbomachine designer a much clearer understanding of the fluid dynamic processes associated with turbomachinery, and thus aids the design process.

Hypersonic propulsion is a subject of increasing engineering interest, particularly the concept of a supersonic combustion ramjet (scramjet), and how it might be implemented in a flight vehicle. The flow field of a two-strut scramjet inlet has been analyzed at Langley Research Center by Kumar, using a three-dimensional Navier-Stokes code.⁶³ The geometry of the inlet is shown in Fig. 4; it has a rectangular cross section with wedge-shaped sidewalls which are swept back at an angle of 30°. Two compression struts are located in the center passage of the inlet; they are also swept back at the same angle. The cowl closure (not shown) begins at the throat. With the aft placement of the cowl, the high-pressure internal flow interacts with the low-pressure external flow in the region ahead of the cowl. This interaction is important to the overall flow characteristics, and must be included in the analysis to accurately predict the inlet flow field. The calculations shown here have included this interaction by extending the flow-field domain below the cowl plane. Pressure contours from a sample calculation in cross-planes of the inlet are shown in Fig. 5. These results were obtained at Mach 4, using a grid consisting of approximately 280,000 points. The pressure contours clearly show the complex wave structure occurring in the inlet.

In order to both improve the reliability and increase the thrust performance of the Space Shuttle main engine (SSME) a large computational simulation research program was undertaken at Ames Research Center by scientists from Rocketdyne and

Ames. A special code was developed that could treat the complex internal geometry of the engine powerhead. Kwak et al.⁶⁴ developed a three-dimensional incompressible Navier-Stokes code (INS3D) for this application that has provided an extensive low-speed simulation capability. Working with engineers from Rocketdyne (the builders of the SSME), this code was applied to simulate the flow within the hot gas manifold, transfer ducts, and main injector the SSME.⁶⁵⁻⁶⁷ Using this code, it was determined that flow within the present three-duct design was inefficient. A large separated flow region existed in the center duct and transmitted only 9% of the total flow. A proposed two-duct design suggested significantly improved flow characteristics. Figure 6 shows the surface-pressure map for the powerhead arrangement. As a result of this computational demonstration, Rocketdyne has determined that CFD simulation will be utilized to develop and analyze all future SSME designs.

Aerospace Applications: External Flows

Researchers at Boeing are utilizing both transonic Euler methods and linear panel methods to study the airframe-propulsion integration effects of aft-mounted ultrabypass ratio engines. Figure 7 shows the surface grid of a complete airplane configuration with aft-mounted ultrabypass engines. This configuration was analyzed at transonic conditions in yaw with actuator disk modeling of the fan using an Euler code by Yu et al.⁶⁸ Figure 8 shows detailed isobars in the aftbody region of the configuration. From solutions such as these the presence of the nacelle and the nacelle-plus-propeller at cruise thrust can be ascertained. In addition, the effects of the nacelle and propeller power on downwash variations in the flow field and on the longitudinal, directional, and lateral stability of the airplane can be studied.

A procedure that employs an unstructured mesh of tetrahedra for treating complex three-dimensional shapes has been combined with a finite-volume (or finite-element) technique by Jameson et al.²⁶ and Jameson and Baker⁶⁹ for solving the Euler equations. This has resulted in a powerful new approach to the problem of calculating flows over complex aircraft. It is capable of predicting realistic flow patterns, provided that adequate computational resources are available. In Fig. 9 the transonic flow results, in the form of pressure contours, are presented for a Boeing 747-200 flying at Mach 0.84 and at an angle of attack of 2.73°. Flow through the engines is permitted but no attempt has been made to simulate the powered engine effects. The flow has been calculated over one half of the aircraft, and the results are reflected about the aircraft plane of symmetry.

A sophisticated composite-grid Euler analysis for the F-16 fighter aircraft has been developed by scientists at General Dynamics, Fort Worth

Division. Their approach utilizes multiple blocks to subdivide a complex problem into several smaller zones that can more accurately represent the geometry and the boundary conditions. Preliminary results for the flow field about the F-16 have been obtained by Karman et al.¹⁸ The grid system is shown in Fig. 10 and typical velocity vectors on the forward fuselage for a Mach 0.9 at 4° angle-of-attack case are shown in Fig. 11. The solution in this region agrees quite well with experimental data. Discrepancies on the upper surface of the wing in the vicinity of the wing shocks are attributed to lack of grid resolution.

Scientists working at the Rockwell International Science Center have developed an Euler procedure for analyzing the flow about multiple bodies.⁷⁰ In this procedure, the physical domain of interest is subdivided into multizones requiring single gridding procedures within each zone. Across zonal boundaries, proper flux balancing is maintained to avoid spurious numerical errors originating at the interface. The zonal interface can be permeable or impermeable and can also be a boundary of flow discontinuity such as a shock wave or sonic surface.

Figure 12 shows the Space Shuttle, solid rocket boosters, and external tank configuration and the resulting inviscid flow-field solution in the form of pressure contours. A five-zone discretization in the axial plane is generated to study this multiple-body problem at supersonic Mach numbers. Pressure contours and the grid are shown at different marching stations for Mach 1.8 and 0° angle of attack.

Modern aircraft wings, including those with tip stores, frequently show pronounced aeroelastic effects. Guruswamy et al.⁷¹⁻⁷³ and Guruswamy and Goorjian⁷⁴ from Ames Research Center have combined a flow-simulation code with a structural response code to develop an aeroelasticity simulation capability. It has been applied to the wings of both the B-1 and F-5. Typical results are shown in Fig. 13, where the pressure distributions for a wing with and without a tip missile are presented.

A new time-accurate Euler method has been developed by Chang at Ames for calculating the transonic flow over a lifting or nonlifting rotor blade in both hover and forward flight.⁷⁵ The procedure solves the conservative Euler equations in a rotor-fixed frame of reference using a finite-volume method. The discretized equations are solved using a Runge-Kutta multistage scheme with a new higher-order implicit residual smoothing procedure. Figure 14 shows the surface pressure contours at three spanwise stations for a lifting, four-bladed rotor in forward flight.

In 1981, recognizing that the building blocks for a three-dimensional Navier-Stokes code existed that could produce a demonstration calculation about a complex configuration, the F-16A fighter was selected as the target for a full three-dimensional transonic computation at Ames Research

Center. A group was formed that simultaneously addressed the geometry, grid development, and flow-solver problems necessary to simulate the flow about this configuration. The key problems were 1) how to divide the flow field into sub-elements that were small enough to be tractable to the existing computer system, 2) how to organize and manage the massive amount of information that would be generated by such a calculation, and 3) how to display the results in a manner usable to the CFD scientist. A Calma CAD/CAM system was used to develop the computer geometry with data supplied by the aircraft manufacturer, the General Dynamics Corporation. Using the Ames ARC3D flow code as a basis, and the concepts of the two-dimensional GRAPE grid generation code, the flow field was divided into 16 separate zones to facilitate the initial computations on the Cray X-MP/22 computer.

Calculations for the F-16 were obtained by Flores et al.⁷⁶ and were performed on the Cray X-MP/48, a larger and faster machine. Typical results from the viscous composite grid code are shown in Figs. 15 and 16. Figure 15 shows the grid system developed for the F-16A, and the flow-field results are shown in Fig. 16. Note the great detail exhibited in the separated-flow regions on the wing, resulting in a simulation that compares quite well with wind-tunnel experiments.

The nature of supersonic flows, with their limited domain of dependence and mathematical description by strictly hyperbolic equations, typically permits an easier solution to be obtained, relative to transonic flows. Specifically, the computational domain is smaller and the shock and boundary-layer interactions are simpler, resulting in faster computations. Hypersonic flows, although incorporating the same advantages as superconic flow, contain additional complications that are yet to be fully evaluated. These include more complicated transition and turbulence physics, real-gas effects, merged shock and boundary layers, stronger shock waves, and radiation effects.

Using a parabolized Navier-Stokes code (PNS), Chaussee et al.⁴³ and Chaussee⁷⁷ developed a flow-simulation capability for supersonic configurations, including the Space shuttle orbiter. This research program was also used to develop the first graphics application on the new Silicon Graphics IRIS work stations. The flow field about the orbiter at reentry conditions is shown in Fig. 17. The development and propagation of vortices are clearly shown, as well as the flow characteristics along the surface. With present work-station capability it is possible for the computational aerodynamicist to carefully examine any selected aspect of the flow field, once a complete solution has been obtained.

Building on the PNS code technology, Rizk et al.⁷⁸ built a combined marching-unsteady code

to simulate hypersonic flows and applied it to a generic hypersonic configuration in support of the National Aerospace Plane (NASP) program. Figure 18 shows the numerically generated particle traces about a generic hypersonic research configuration at Mach 25 and at an angle of attack of 5°. Additional flow-field realism is planned by including real-gas chemistry effects in the computations.

Fluid Physics Applications

Flows around bodies and wings at high angles of attack have assumed increasing importance in recent years as flight vehicles such as highly maneuverable fighter aircraft and the Space Shuttle operate in this flight regime. High-angle-of-attack flows are difficult to understand and compute. One of the principal reasons is the existence of large, vortex-dominated flow separations with complicated flow physics that are not yet well understood. A few selected examples will demonstrate some of the complexities and results that have recently been obtained.

On a slender wing at moderate to high angles of attack, the flow separates from the leading edge and forms two spiral shaped vortices above the upper surface of the wing. These separation vortices induce low pressure on the upper surface, which is the key factor in the nonlinear lift increase in the fighter-type wing. At large angles of attack, breakdown of the lee vortices occurs near the leading edge. The vortex breakdown phenomenon causes both a loss in the lift and a nose-up pitching moment. For modern fighter aircraft, strakes are often added to augment and control the lift at high angles of attack. A feature of such a configuration is the interaction between the vortices emanating from the strake and wing leading edges. Although the vortex interaction and vortex breakdown are important flow characteristics, neither mechanism is presently well understood. Current computational research is leading to enhanced understanding, and eventual control of vortex behavior.

Viscous separated flows surrounding a hemisphere cylinder at angles of attack ranging up to 19° in transonic flow have been computed by Ying et al.⁷⁹ at Ames using an implicit, approximately factored, partially flux-split algorithm. The resulting flow-field structures, including the vortical flow on the lee of the body and the three-dimensional separation patterns, have been investigated. The computed results show good qualitative and quantitative agreement with experimental data. The computed surface singular-point topology is in agreement with that inferred from the oil-flow results. A typical result depicting the computed three-dimensional particle traces at Mach 0.9, angle of attack of 19°, and a Reynolds number based on body diameter of 212,500 is shown in Fig. 19. In addition this same code was also

used to study the fluid physics of rapid pitch-up of the hemisphere cylinder at Mach 1.2.

The viscous flow about a planar delta wing at angle of attack has been computed by Thomas et al.⁴ at Langley, using an efficient algorithm for the compressible Navier-Stokes equations. Figure 20 shows the total pressure contours for a highly swept delta wing at Mach 0.3, an angle of attack of 20.5° , and a Reynolds number based on length of 9.5×10^5 . The contours indicate the primary vortex, which is shed from the leading edge, and the secondary vortices (underneath the primary), which are induced by boundary-layer cross-flow separation. The results showed good agreement with experimental data and include the prediction of maximum lift associated with the onset of vortex breakdown.

The computed flow fields over a strake-delta wing exhibiting vortex breakdown obtained by Fujii et al.⁸⁰ are shown in Figs. 21 and 22. The flow simulations were carried out on the Cray-2 computer, using a three-dimensional Navier-Stokes code, at flow conditions of Mach 0.3, angle of attack of 30.0° , and Reynolds number of 1.3×10^6 . Figure 21 (original in color) shows the total-pressure contour plots in cross-flow planes. The position of the vortex cores are characterized by the low total pressure denoted by yellow or green. As the total pressure approaches free-stream value, the color becomes red to pink. It is observed that the primary vortex is much more tightly concentrated, except near the trailing edge, where the core size suddenly spreads. This abrupt change of the flow is induced by the pressure increase toward the trailing edge, and signifies vortex breakdown. Figure 22 (original in color) shows the corresponding particle traces. Red particles show the vortex emanating from the strake region, and green particles show the vortex from the wing region. Yellow particles are released in the vortical region at the trailing edge, and clearly show the flow reversal associated with vortex breakdown. It is also noticed that the strake vortex is moved outward and downward aft of the strake-wing junction, a result of the interaction with the wing vortex.

The formation and propagation of turbulence are not well understood and are very difficult to predict. It is very important to be able to model the behavior of turbulence for use in Reynolds-averaged Navier-Stokes codes. With modern CFD software simulation tools in conjunction with the latest computer systems, it is now becoming possible to computationally simulate turbulent flows (i.e., for simple shapes and low Reynolds numbers). The program at Ames is the most advanced in the nation and possibly in the world. Both large-eddy simulation (LES) and direct-simulation (DS) techniques are employed, with computations on three-dimensional grids taking of the order of 100 hr of computer time. A typical result from an LES calculation by Kim and Moin^{81,82} is shown in Fig. 23, which depicts a

horseshoe vortex. In this calculation, inverted horseshoe vortices were first discovered that were subsequently verified by experimentalists.

Direct numerical simulations of turbulent spots in plane Poiseuille and boundary-layer flows have been performed by Henningson et al.⁸³ at Ames. The Poiseuille case was calculated at a Reynolds number of 1,500 (based on the laminar centerline velocity and channel half-height) and the spot was followed about 100 channel heights downstream. The boundary-layer spot was started at a Reynolds number of 250 (based on the initial displacement thickness and the free-stream velocity) and was followed about 200 initial boundary-layer thicknesses downstream. The propagation velocities and spreading angles were found to compare well with corresponding experiments. The difference in shape of the two spots was clearly discernible: the turbulent parts were contained within arrow-head regions that point in opposite directions for the two cases. Figure 24 shows the turbulent spot for the boundary-layer case. The wingtip region of the Poiseuille spot was also found to consist of a large-amplitude, semiturbulent wave packet.

Computer Technology

CFD is a highly computer-intensive discipline that critically depends on the availability of advanced computers. The usual driver is supercomputer technology. More recently with the increased use of work stations and networks greater attention has been given to the entire supercomputing environment.⁸⁴ This section expands beyond supercomputer technology to include three additional technology areas: scientific work stations, local supercomputing networks, and remote access.

Supercomputers

CFD's increasing demands for higher and higher performance and capacity can only be satisfied by supercomputer-class systems. These systems are usually defined as those providing the highest floating-point calculation rate and largest memory capacity. By definition, supercomputers are at the leading edge of technology and a given system loses its "super" status when a newer technology arrives on the scene. In contrast to the late 1960s and most of the 1970s, when CFD was in its early stages and there were only a few supercomputers, the 1980s have seen a renewed interest in supercomputers both by the computer vendors and by the scientific community. This interest is at least partly a result of the recognition by the government, industry, and universities that supercomputers provide an essential tool for advancing the nation's research and development efforts. In some instances, such as the National Aerospace Plane Program, supercomputers provide an essential and enabling technology. Recognition of the importance of

supercomputers has also been recognized abroad, particularly in Japan. Japanese supercomputers have been built by Fujitsu, Hitachi, and NEC. These systems, as well as major U.S. products, are described in Ref. 85. There also is a Japanese National Supercomputer Project with the goal of achieving 10 billion floating-point operations per second before the end of the decade.

The growth in supercomputer performance in recent years is indeed impressive. This is illustrated in Fig. 25 where the actual and anticipated performance of selected Control Data Corp., Cray Research, Inc., and ETA Systems, Inc., supercomputers is plotted as a function of the year of introduction. Equally impressive is the comparative slow price growth (Fig. 26). These two figures indicate that whereas performance has increased by a factor of 10,000 in less than 25 years, the purchase price has increased by a factor of only 5.

The continuous growth in supercomputer performance is due to several factors. Hardware improvements, including faster switching devices, denser semiconductor chips, and improved packaging, account for much of the improvement. The improvement in machine cycle time, shown in Fig. 27, is one measure of performance increases owing to hardware improvements. However, comparing Fig. 27 and Fig. 25 shows that hardware advances account for only about half the performance improvement seen over the past 25 years. In fact, the rate of improvement has slowed in recent years. The Cray-3, which is anticipated to arrive in 1988 or 1989, and the Cray-MP which is expected in 1990 or 1991 are projected to have a 2-nsec clock cycle. This is only half the 4-nsec clock cycle of the Cray-2, which was introduced in 1985. The faster clock cycle of the Cray-3 can only be achieved by replacing silicon with gallium arsenide. Further reduction to a 1-nsec clock will require high electron mobility technology (HEMT), a modified version of gallium arsenide still in the laboratory. Other exotic technologies including fiber optics and high-temperature superconductors are sure to be investigated for application to supercomputers. As switching speeds become faster, the signal paths within the supercomputer must be shortened to prevent long signal delays between chips. Consequently, the entire computer shrinks, which places a significant burden on cooling and packaging technologies. Already, exotic cooling techniques such as the Cray-2 fluorocarbon immersion and the ETA-10 liquid nitrogen immersion have been used.

To achieve today's high supercomputer performance, the advanced hardware technologies have had to be supplemented by advanced architectures began in the mid-1970s with the introduction of the ILLIAC IV parallel processor and the STAR-100 vector processor. Although neither was a success in the marketplace they established a direction for supercomputers that has been followed ever since. Recently, the Cray X-MP and the Cray-2

have introduced powerful multivector processor architectures. Future architectures will surely expand this organization to include even more processors. For example, the ETA-10 and Cray Y-MP are purported to have eight processors, the Cray-3 to have 16, and the Cray-MP to be organized into clusters of 16 processors each. These are examples of architectures consisting of a relatively few very powerful processors sharing a common memory. In contrast, there are many examples, such as the Intel Hypercube, of organizations consisting of hundreds, and possibly thousands, of processors, each with its own piece of a distributed memory. For the near term the former organization appears the most likely for supercomputers, but the potential for the massively parallel processor design is clearly evident. In any case, significant increases in supercomputer performance will come from parallel processing.

The future direction of supercomputer architectures will not be determined solely by hardware technology and exotic organization, but rather by the ease with which the scientific user can program, debug, and execute his problem. The effective application of multiprocessor architectures to CFD and other scientific problems is probably the single greatest challenge in scientific computing. When one notes that it has taken the scientific community approximately 10 years to fully exploit vector computers, the programming of multiprocessors needs to be addressed immediately. Issues that need to be addressed include programming languages, code debugging tools, and, perhaps the most challenging of all, new algorithms. Efforts are, of course, being made in all these areas and it is to be hoped they will bear fruit before the new supercomputers arrive. It may, by the way, turn out that new architectures are so difficult to utilize for the general mix of problems that the CFD researcher will use a loosely coupled network of special supercomputers, each with an architecture suited to a particular class of applications.

The new parallel architectures could not exist without advanced high-density semiconductors; only these advanced components offer the short path lengths and reliability required by such complex machines. High-density semiconductors have also provided a great increase in supercomputer memory capacity. The past and future growth in memory capacity is shown in Fig. 28. In the past, memory size increased steadily, but supercomputers were capacity-limited. Usually, the processor quickly consumed central memory storage and a large fraction of time was spent in input/output to secondary disk storage. Today, much larger memory capacities, made possible by designs that take advantage of the slower but much denser 256 kb dynamic memory chip, are available. The Cray-2, which has 268,435,456 words (64-bit) (this is often denoted as 256 megawords where mega = $(1024)^2$) of central memory, is the first supermemory supercomputer. At the time of its introduction in 1985 the Cray-2

central memory capacity was greater than the combined capacity of the approximately 100 then installed Cray-1 and Cray X-MP computers. Super-memory supercomputers are here to stay and memories can be expected to grow to 1 billion and then to 4 billion words within the next 5 years. Even more significantly, the availability of large memory has afforded the CFD researcher the capability to solve problems that were intractable on smaller-memory machines.

Work Stations

The scientific work station is a revolutionary new computer technology that is quickly finding its place in CFD. The modern work station has grown out of the early intelligent terminals and vector display devices of the 1970s; today it has surpassed both to provide powerful processing and raster display capabilities. Figure 29 shows the projection in work-station floating-point performance as compared to the Digital Equipment Corp. VAX 11/780. The Motorola 68020 in combination with the Weitek IEEE floating-point chip matches the VAX 11/780 performance, and the new reduced instruction set computer (RISC) chips will allow new work stations to greatly exceed that of the standard department-level computer of just a few years ago.

Whereas the general-purpose performance of scientific work stations brings rapid response to the individual user for such tasks as text editing, report preparation, code module debugging, and small test-code execution, their major benefit has been the ability to provide powerful graphical data display and analysis capabilities.⁵⁸ It is universally agreed that the massive data generated by modern CFD models can only be assimilated by graphical techniques. The scientific work-station operational paradigm brings graphical data directly to the individual research's desktop display where it can be viewed from different perspectives, enlarged, or clipped in real time, and then saved on videotape or film for later viewing.^{86,87} The projected rapid increase in scientific work-station raster graphic transformation and display capabilities in terms of display rates for both coordinate points and 100-pixel flat, shaded polygons is given in Fig. 30. The two-order of magnitude increase in performance is needed for good motion viewing of complex solid objects like aircraft. When work stations are not sufficiently powerful, it may be necessary to use the supercomputer. This is now possible, and with the supercomputing network described below, one can perform interactive graphics on the supercomputer just as if it were a work station.^{88,89}

Supercomputing Networks

The sophistication of CFD has increased to the degree that a relatively complex and diverse collection of hardware and software systems is needed for its support. On the hardware side, the

trend is away from the monolithic general-purpose computer or the classic supercomputer back-end/mainframe-computer front-end arrangement. In its place is a local-area network of supercomputers, mainframe computers, minicomputers, and microcomputers. The trend toward supercomputing networks is influenced by several factors. There is the need to share capital-intensive systems such as supercomputers and mass storage systems over a large number of users. There is the need to increase reliability by minimizing the effect of a single system failure. There is the advantage of modularity when implementing upgrades and other improvements. Finally, the user can partition his work to take optimal advantage of specific hardware and software features of particular systems in the network.

The supercomputing network consists of hardware systems from different vendors, and if the system is to keep pace with advanced technologies, it must be designed to accommodate new vendors. That is, it is important to implement a supercomputing system that has the flexibility to readily accommodate new technology. It is even more important that disruption to the CFD user is minimized each time the system is upgraded. Furthermore, the differences between vendor products as seen by the user must be minimized. Nevertheless, the trend toward networks of computers raises the potential of forcing the user to use a multitude of operating systems. The software systems staff is also faced with the problems of getting these different computers to exchange data. On the other hand, the opportunity to provide a powerful supercomputing system rich in function and performance yet uniform in software and user interface is too enticing to ignore. Recently, developments toward reaching this goal have taken place.

An important development in providing a uniform user-view is the network-wide implementation of open-system architectures. An open-system architecture is a hardware or software specification that is available for general implementation. Notable examples include AT&T's UNIX operating system, Department of Defense's Internet Protocol (commonly called TCP/IP), and Xerox's Ethernet. The advantage of these open-system architectures is that they are implemented on nearly all computer systems and by several vendors. They have been used, for example, by the NAS program to implement a supercomputing network offering a common UNIX operating system and network software environment across six different vendor's computer systems.^{90,91}

Another important development that has increased the functional richness of supercomputing is interactive supercomputer operating systems. The interactive operating mode that users have enjoyed on other systems is rapidly entering the supercomputer arena with the increasing use of CTSS (Cray time-sharing system) and Cray's introduction of UNICOS. Now supercomputers can be viewed as fully participating hosts in the

supercomputing network. In this environment valuable tools such as supercomputer/work-station interactive graphics processing become routine.⁸⁹

Future systems will further expand the supercomputing network model to incorporate distributed file systems and distributed operating systems in such a manner that the CFD user will not be concerned about the details of which particular computer is performing each portion of this task. Computing systems are heading toward even greater levels of transparency with distributed operating and file systems that automatically assign data and processing to available resources. The user will truly see the system as just a single resource.

Remote Access

The recent increase in supercomputer installations at government, industry, and university sites has increased the requirement for supercomputer access by remote users. The National Science Foundation (NSF), for example, is implementing a national supercomputer network by connecting NSF supercomputer and other regional networks to a transcontinental backbone circuit. The National Magnetic Fusion Energy Computer Center has had a network, MFEnet, for some years, and the Numerical Aerodynamic Simulation (NAS) Program has implemented NASnet for industrial and NASA sites.^{90,91}

The ultimate goal of remote access networks is to provide access at a consistent high quality of service to users regardless of their location. Access and service are defined to mean that a user is able to input data, receive output, and in general interact with a host system whether it is across the street or across the country. For example, NASnet uses communication bridges to connect remote Ethernets to the NAS local Ethernet via 56-kb/sec terrestrial circuits. This connection, together with the TCP/IP protocol, allows access from a great variety of large and small computers. Work-station clusters have been implemented at both Langley and Lewis research centers providing these users the same capabilities, including interactive graphics, as the users at Ames.

Functional capabilities that are independent of location are important but not sufficient. Higher performance will be necessary for access to future supercomputers. This is true even if it is not necessary that all data be shipped to or from the user via the communication links. Only the data required to perform the necessary task must be transferred. If it is assumed, for example, that very large archival data files can be sent to the user by means of transportable media such as magnetic tape, then the transport of graphical data becomes the network performance driver. Graphic display bandwidth at movie rates is about 1 Gb/sec. Today there are only a few circuits at 1.544 Mb/sec and several at 56 kb/sec; but,

because of high cost, most are at 9.6 kb/sec. However, this will change significantly with the introduction of the new fiber-optics technology. Fiber-optic circuits now being implemented throughout the country offer much higher bandwidths and also, barring overregulation and restrictive tariffs, much lower cost. In addition, new communication approaches utilizing satellite broadcast video are possible. In this case the user gives commands to a remote graphics processor over fiber-optic links and receives display images via satellite video transmissions. In summary, both requirements and technologies exist for a revolution in worldwide high-speed data communications. The future CFD researcher, regardless of his location, will have ready access to the best supercomputing capabilities anywhere in the country directly from his desktop computer.

Concluding Remarks

The theory, applications and computer technology discussed above depict the current capability of CFD in this country to simulate the realistic flow about some rather complicated aerospace vehicles and their components. With improvements in both the computational simulation tools and computer technology, more sophisticated calculations will be performed. These will eventually find their way into the design-cycle process and lead to improved aerospace vehicle performance at a reduced development cost.

The future of fluid dynamics, both computational and experimental, is bright; it offers many challenges and should produce some advances not only in CFD but also in fluid physics. It is believed that most problems amenable to solution using validated CFD tools can be solved if the desire and resources are devoted to them.

Computational fluid dynamics with its basic objectives of providing a tool to understand and discover fluid flow phenomena, to supplement or complement experimental testing, and to aid the aerospace vehicle design process, has had and will continue to have a tremendous effect on aerodynamic simulation.

References

¹"National Aeronautical R&D Goals," Aeronautical Policy Review Committee, Executive Office of the President, Office of Science and Technology Policy, Washington, D.C., Mar. 1985.

²Shang, J. S., "An Assessment of Numerical Solutions of the Compressible Navier-Stokes Equations," *AIAA Journal of Aircraft*, Vol. 2, No. 5, May 1985, pp. 353-370.

- ³Holst, T. L., "Numerical Solution of the Navier-Stokes Equations about Three-Dimensional Configurations--A Survey," presented at the NASA Conference on Supercomputing in Aerospace, Moffett Field, Calif., Mar. 1987.
- ⁴Thomas, J. L., Taylor, S. L., and Anderson, W. K., "Navier-Stokes Computations of Vortical Flows over Low Aspect Ratio Wings," AIAA Paper 87-0207, Reno, Nev., 1987.
- ⁵Pan, D. and Pulliam, T. H., "The Computations of Steady Three-Dimensional Separated Flows over Aerodynamic Bodies at Incidence and Yaw," AIAA Paper 84-0109, 1986.
- ⁶Ying, S. X., Steger, J. L., Schiff, L. B., and Baganoff, D., "Numerical Simulation of Unsteady, Viscous, High Angle of Attack Flows Using a Partially Flux-Split Algorithm," AIAA Paper 86-2179, 1986.
- ⁷Chaderjian, N. M., "Transonic Navier-Stokes Wing Solutions Using a Zonal Approach. Part 2. High Angle of Attack Simulation," AGARD 58th Fluid Dynamics Panel Symposium, Aix-en-Provence, France, Apr. 1986.
- ⁸Kaynak, U., Holst, T. L., and Cantwell, B. J., "Computation of Transonic Separated Wing Flows Using an Euler/Navier-Stokes Zonal Approach," NASA TM-88311, 1986.
- ⁹Newsome, R. W. and Adams, M. S., "Numerical Simulation of Vortical-Flow over an Elliptical-Body Missile at High Angles of Attack," AIAA Paper 86-0559, 1986.
- ¹⁰Shang, J. S. and Scherr, S. J., "Navier-Stokes Solution of the Flow Field around a Complete Aircraft," AIAA Paper 85-1509, 1985.
- ¹¹"Current Capabilities and Future Directions in Computational Fluid Dynamics," NRC Committee, R. G. Bradley, Chairman, National Academy Press, Washington, D.C., 1986.
- ¹²Rogallo, R. S. and Moin, P., "Numerical Simulation of Turbulent Flows," Annual Review of Fluid Mechanics, Vol. 16, 1984, pp. 99-137.
- ¹³Hussaini, M. Y., "Stability, Transition and Turbulence," Proceedings of the Conference on Supercomputing in Aerospace, edited by P. Kutler and H. Yee, NASA CP-2454, 1987.
- ¹⁴Rubesin, M. W., "Turbulence Modeling," Proceedings of the Conference on Supercomputing in Aerospace, edited by P. Kutler and H. Yee, NASA CP-2454, 1987.
- ¹⁵Reynolds, W. C., "Advances in Turbulence Physics and Modeling by Direct Numerical Simulations," Proceedings of the Conference on Supercomputing in Aerospace, edited by P. Kutler and H. Yee, NASA CP-2454, 1987.
- ¹⁶Dyower, D. L. and Kumar, A., "Computational Analysis of Hypersonic Airbreathing Aircraft Flow Fields," Proceedings of the Conference on Supercomputing in Aerospace, edited by P. Kutler and H. Yee, NASA CP-2454, 1987.
- ¹⁷Howe, J. T., "Introductory Aerothermodynamics of Advanced Space Transportation Systems," AIAA Journal of Spacecraft, Vol. 22, No. 1, Jan.-Feb. 1985.
- ¹⁸Karman, S. L., Jr., Steinbrenner, J. P., and Kislewski, K. M., "Analysis of the F-16 Flow Field by a Block Grid Euler Approach," AGARD 58th Fluid Dynamics Panel Symposium, Aix-en-Provence, France, Apr. 1986.
- ¹⁹Lombard, C. K. and Venkatapathy, E., "Implicit Boundary Treatment for Joined and Disjoint Patched Mesh Systems," AIAA Paper 85-1503-CP, 1985.
- ²⁰Benek, J. A., Buning, P. G., and Steger, J. L., "A 3-D Chimera Grid Embedding Technique," AIAA Paper 1523, Cincinnati, Ohio, 1985.
- ²¹Szema, K. Y. et al., "Multi-Zone Euler Marching Technique for Flow over Single and Multi-Body Configurations," AIAA Paper 87-0592, 1987.
- ²²Rai, M. M., "Navier-Stokes Simulations of Rotor-Stator Interaction Using Patched and Overlaid Grids," AIAA Paper 85-1519, Cincinnati, Ohio, 1985.
- ²³Rai, M. M., "A Conservative Treatment of Zonal Boundaries for Euler Equation Calculations," AIAA Paper 84-0164, 1984.
- ²⁴Flores, J., Holst, T. L., Kaynak, U., Gundy, K., and Thomas, S. D., "Transonic Navier Stokes Wing Solution Using a Zonal Approach. Part 1. Solution Methodology and Code Validation," AGARD 58th Fluid Dynamics Panel Symposium, Aix-en-Provence, France, Apr. 1986.
- ²⁵Yu, N. J., Kusnose, K., Chen, H. C., and Sommerfield, D. M., "Flow Simulations for a Complex Airplane Configuration Using Euler Equations," AIAA Paper 87-0454, 1987.
- ²⁶Jameson, A., Baker, T. J., and Weatherhill, N. P., "Calculation of Inviscid Transonic Flow over a Complete Aircraft," AIAA Paper 86-103, Jan. 1986.
- ²⁷Pulliam, T. H. and Steger, J. L., "On Implicit Finite Difference Simulations of Three-Dimensional Flows," AIAA Journal, Vol. 18, Feb. 1979.
- ²⁸Pulliam, T. H. and Steger, J. L., "Recent Improvements in Efficiency, Accuracy, and Convergence of an Implicit Approximate Factorization Algorithm," AIAA Paper 85-0360, 1985.

- 29 Jameson, A., Schmidt, W., and Turkel, E., "Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes," AIAA Paper 81-1259, 1981.
- 30 McCormack, R. W., "A Numerical Method for Solving the Equations of Compressible Viscous Flow," AIAA Paper 81-0110, 1981.
- 31 Thomas, J. L. and Walters, R. W., "Upwind Relaxation Algorithms for the Navier-Stokes Equations," AIAA Paper 85-1501-CP, 1985.
- 32 Chakravarthy, S. R. and Osher, S., "Numerical Experiments with the Osher Upwind Scheme for the Euler Equations," AIAA Paper 82-0975, 1982.
- 33 Chakravarthy, S. R., "Relaxation Methods for Unfactored Schemes," AIAA Paper 84-0165, 1984.
- 34 Anderson, W. K., Thomas, J. L., and Van Leer, B., "A Comparison of Finite Volume Flux Vector Splittings for the Euler Equations," AIAA Paper 85-0122, 1985.
- 35 McCormack, R. W., "Current Status of Numerical Solutions of the Navier-Stokes Equations," AIAA Paper 85-0032, 1985.
- 36 Yee, H. C., Warming, R. F., and Harten, A., "Implicit Total Variation Diminishing (TVD) Schemes for Steady-State Calculations," NASA TM-84342, 1983.
- 37 Yee, H. C. and Harten, A., "Implicit TVD Schemes for Hyperbolic Conservation Laws in Curvilinear Coordinates," AIAA Paper 85-1513, 1985.
- 38 Jespersen, D. C., "Recent Developments in Multigrid Methods for the Steady Euler Equations," Lecture Notes for Lecture Series on Computational Fluid Dynamics, von Karman Institute for Fluid Dynamics, Belgium, 1984.
- 39 Jameson, A., "Solution of the Euler Equations for Two-Dimensional Transonic Flow by a Multigrid Method," Applied Mathematics and Computation, Vol. 13, 1983, pp. 87-110.
- 40 Turkel, E., "Preconditioned Methods for Solving the Incompressible and Low Speed Compressible Equations," NASA ICASE Report No. 86-14, 1986.
- 41 Jespersen, D. C. and Buning, P. G., "Accelerating an Iterative Process by Explicit Annihilation," NASA TM-84304, 1983.
- 42 Wigton, L. B., Yu, N. J., and Young, D. P., "GMRES Acceleration of Computational Fluid Dynamics Codes," AIAA Paper 85-1494, 1985.
- 43 Chaussee, D. S., Rizk, Y. M., and Buning, P. G., "Viscous Computation of a Space Shuttle Flow Field," Ninth International Conference on Numerical Methods in Fluid Dynamics, Saclay, France, June 1984. (See also NASA TM-85977, 1984.)
- 44 Chakravarthy, S. R. and Szema, K. Y., "An Euler Solver for Three-Dimensional Supersonic Flows with Subsonic Pockets," AIAA Paper 85-1703, 1985.
- 45 Rubesin, M. W. and Viegas, J. R., "A Critical Examination of the Use of Wall Functions as Boundary Conditions in Aerodynamic Calculations," Third Symposium on Numerical and Physical Aspects of Aerodynamic Flows, California State University, Long Beach, Calif., Jan. 1985.
- 46 Van Dalsem, W. R. and Steger, J. L., "The Fortified Navier-Stokes Approach," Proceedings of the Workshop on Computational Fluid Dynamics, Institute of Nonlinear Sciences, University of California, Davis, Calif., June 17-18, 1986.
- 47 Chaussee, D. S. and Pulliam, T. H., "A Diagonal Form of an Implicit Approximate Factorization Algorithm with Application to a Two Dimensional Inlet," AIAA Journal, Vol. 19, No. 2, Feb. 1981.
- 48 Barth, T. J. and Steger, J. L., "An Efficient Approximate Factorization Implicit Scheme for the Equations of Gasdynamics," NASA TM-85957, 1984.
- 49 Coakley, T., "Numerical Method for Gas Dynamics Combining Characteristics and Conservation Concepts," AIAA Paper 81-1257, 1981.
- 50 Johnson, F. T., Bussoletti, J. E., Woo, A. C., and Young, D. P., "A Transonic Rectangular Grid Embedded Panel Method," Recent Advances in Numerical Methods in Fluids, Vol. 4, edited by W. G. Habashi, Pineridge Press, Ltd., 1984.
- 51 Samant, S. S., Bussoletti, J. E. et al., "TRANAIR: A Computer Code for Transonic Analyses of Arbitrary Configurations," AIAA Paper 87-0034, Reno, Nev., 1987.
- 52 Hafez, M. M., Habashi, W. G., Przybytkowski, S. M., and Peeters, M. F., "Compressible Viscous Internal Flow Calculations by a Finite Element Method," AIAA Paper 87-0644, 1987.
- 53 Grossman, B., "The Computation of Inviscid Rotational Gasdynamic Flows Using an Alternate Velocity Decomposition," AIAA Paper 83-1900, 1983.
- 54 Chaderjian, N. M. and Steger, J. L., "The Numerical Simulation of Steady Transonic Rotational Flow Using a Dual Potential Formulation," AIAA Paper 85-0368, 1985.
- 55 Rao, K. V., Steger, J. L., and Fletcher, R. H., "A Three-Dimensional Dual Potential Procedure for Inlets and Indraft Wind Tunnels," AIAA Paper 87-0598, 1987.
- 56 Davis, R. L., Carter, J. E., and Hafez, M. M., "Three-Dimensional Viscous Flow Solutions with a Vorticity-Stream Function Formulation," AIAA Paper 87-0601, 1987.

- 57 Hafez, M. and Lovell, D., "Entropy and Vorticity Corrections for Transonic Flows," AIAA Paper 83-1926, 1983.
- 58 Buning, P. G. and Steger, J. L., "Graphics and Flow Visualization in Computational Fluid Dynamics," AIAA Paper 85-1507-CP, Proceedings of the AIAA 7th Computational Fluid Dynamics Conference, 1985.
- 59 Andrews, A. E., "Progress and Challenges in the Applications of Artificial Intelligence to Computational Fluid Dynamics," AIAA Paper 87-0593, 1987.
- 60 Holst, T. L., Kaynak, U. et al., "Numerical Solution of Transonic Wing Flows Using an Euler/Navier-Stokes Zonal Approach," AIAA Paper 85-1640, 1985.
- 61 Chima, R. V., "Development of an Explicit Multi-Grid Algorithm for Quasi-Three-Dimensional Viscous Flows in Turbo Machinery," NASA TM-87128, 1986.
- 62 Rai, M. M., "A Relaxation Approach to Patched-Grid Calculations with the Euler Equations," Journal of Computational Physics, Vol. 66, No. 1, Sept. 1986, pp. 99-131.
- 63 Kumar, A., "Numerical Simulation of Scramjet Inlet Flow Fields," NASA TP-2517, 1986.
- 64 Kwak, D., Chang, J. L. C., Shanks, S. P., and Chakravarthy, S., "A Three-Dimensional Incompressible Navier-Stokes Flow Solver Using Primitive Variables," AIAA Journal, Vol. 24, No. 3, Mar. 1986, pp. 390-396.
- 65 Chang, J. L. C., Kwak, D., Dao, S. C., and Rosen, R., "A Three-Dimensional Incompressible Flow Simulation Method and Its Application to the Space Shuttle Main Engine. Part I. Laminar Flow," AIAA Paper 85-0175, Reno, Nev., 1985.
- 66 Chang, J. L. C., Kwak, D., Dao, S. C., and Rosen, R., "A Three-Dimensional Incompressible Flow Simulation Method and Its Application to the Space Shuttle Main Engine. Part II. Turbulent Flow," AIAA Paper 85-1670, Cincinnati, Ohio, 1985.
- 67 Yang, R.-J., Chang, J. L. C., and Kwak, D., "A Navier-Stokes Simulation of the Space Shuttle Main Engine Hot Gas Manifold," AIAA Paper 87-0368, Reno, Nev., 1987.
- 68 Yu, N. J., Kusunose, K., Chen, H. C., and Sommerfield, D. M., "Flow Simulations for a Complex Airplane Configuration Using Euler Equations," AIAA Paper 87-0454, Reno, Nev., 1987.
- 69 Jameson, A. and Baker, T. J., "Improvements to the Aircraft Euler Method," AIAA Paper 87-0452, Reno, Nev., 1987.
- 70 Shankar, V. and Chakravarthy, S., "Development and Application of Unified Algorithms for Problems in Computational Science," Supercomputing in Aerospace, NASA CP-2454, 1987.
- 71 Guruswamy, G. P., Goorjian, P. M., and Tu, E. L., "Transonic Aeroelasticity of Wings with Tip Stores," Paper 86-1007-CP, AIAA/ASME/ASCE/AHS 27th Structures, Structural Dynamics and Materials Conference, San Antonio, Tex., 1986.
- 72 Guruswamy, P., Goorjian, P. M., Ide, H., and Miller, G. D., "Transonic Aeroelastic Analysis of the B-1 Wing," Journal of Aircraft, Vol. 23, No. 7, July 1986, pp. 547-553.
- 73 Guruswamy, G. P., Goorjian, P. M., and Tu, E. L., "Unsteady Transonics of a Wing with Tip Stores," Journal of Aircraft, Vol. 23, No. 8, Aug. 1986, pp. 662-668.
- 74 Guruswamy, P. and Goorjian, P. M., "Efficient Algorithm for Unsteady Transonic Aerodynamics of Low-Aspect-Ratio Wings," Journal of Aircraft, Vol. 22, No. 3, Mar. 1985, pp. 193-199.
- 75 Chang, I.-Chung, "Unsteady Euler Solution of Transonic Helicopter Rotor Flow," American Helicopter Society, National Specialists' Meeting on Aerodynamics and Aeroacoustics, Arlington, Tex., 1987.
- 76 Flores, J., Reznick, S. G., Holst, T. L., and Gundy, K., "Transonic Navier-Stokes Solutions for a Fighter-Like Configuration," AIAA Paper 87-0032, Reno, Nev., 1987.
- 77 Chaussee, D. S., "High Speed Viscous Flow Calculations about Complex Configurations," Paper 29, 58th Meeting of the Fluid Dynamics Panel Symposium on Applications of Computational Fluid Dynamics in Aeronautics, Aix-en-Provence, France, Apr. 1986.
- 78 Rizk, Y., Chaussee, D., and Steger, J., "Numerical Simulation of the Hypersonic Flows around Lifting Vehicles," Symposium on the Aerodynamics of Hypersonic Lifting Vehicles Sponsored by the AGARD Fluid Dynamics Panel, to be held in Bristol, United Kingdom, Apr. 6-9, 1987.
- 79 Ying, S. X., Steger, J. L., and Schiff, L. B., "Numerical Simulation of Unsteady, Viscous, High-Angle-of-Attack Flows Using a Partially Flux-Split Algorithm," AIAA Paper 86-2179, Williamsburg, Va., 1986.
- 80 Fujii, K., Van Dalsem, W. R., Schiff, L. B., and Steger, J. L., "Numerical Simulation over a Strake-Delta Wing," AIAA Paper 87-1229, Honolulu, Hawaii, 1987.
- 81 Moin, P. and Kim, J., "The Structure of the Vorticity Field in Turbulent Channel Flow. Part I. Analyses of Instantaneous Fields Statistical Correlations," Journal of Fluid Mechanics, No. 155, 1985, pp. 441-464.

⁸²Kim, J. and Moin, P., "The Structure of the Vorticity Field in Turbulent Channel Flow. Part II. Study of Ensemble-Averaged Fields," Journal of Fluid Mechanics, No. 162, 1986, pp. 339-363.

⁸³Henningson, D., Spalart, P., and Kim, J., "Numerical Simulations of Turbulent Spots in Plane Poiseuille and Boundary-Layer Flow," submitted to Physics of Fluids.

⁸⁴Bailey, F. R., "Scientific Computing Environment for the 1980s," Numerical Methods for Engine-Airframe Integration, edited by S. N. B. Murthy and G. C. Paynter, AIAA Progress in Astronautics and Aeronautics, Vol. 102, 1986, pp. 3-22.

⁸⁵Fernbach, S., ed., Supercomputers Class VI Systems, Hardware and Software, Elsevier Science Publishing Co., New York, N.Y., 1986.

⁸⁶Lasinski, T. et al., "Flow Visualization of CFD Using Graphics Workstations," AIAA Paper 87-1180-CP, June 1987.

⁸⁷Watson, V., Buning, P., and Choi, D., "Use of Computer Graphics for Visualization of Flow Fields," AIAA Aerospace Engineering Conference and Show, Los Angeles, Calif., 1987.

⁸⁸Choi, D. and Levit, C., "An Implementation of a Distributed Interactive Graphics System for a Supercomputer Environment," International Journal of Supercomputing Applications, 1987.

⁸⁹Rogers, S. E., Buning, P. G., and Merritt, F. J., "Distributed Interactive Graphics Applications in Computational Fluid Dynamics," International Journal of Supercomputing Applications, 1987.

⁹⁰Bailey, F. R., "Status and Projections of the NAS Program," Computational Mechanics--Advances and Trends--AMD, Vol. 75, The American Society of Mechanical Engineers, 1986.

⁹¹Bailey, F. R., "NAS--Current Status and Future Plans," Supercomputing in Aerospace, NASA CP-2454, 1987.

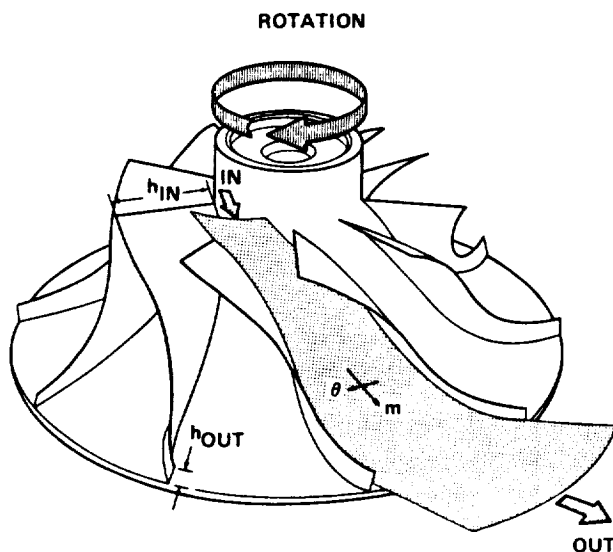


Fig. 1 Impeller geometry.

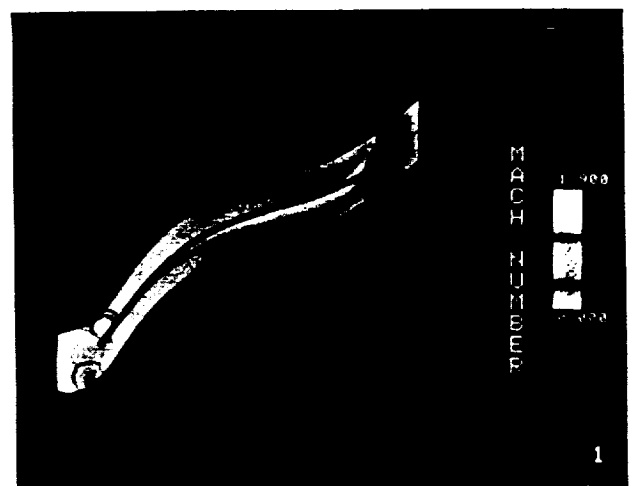


Fig. 2 Relative Mach-number contours.

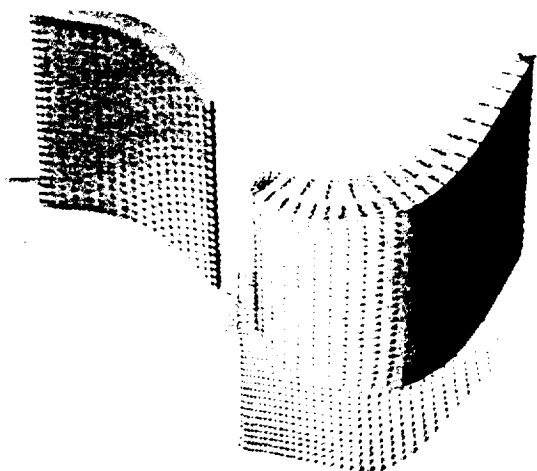


Fig. 3 Three-dimensional rotor-stator velocity vectors.

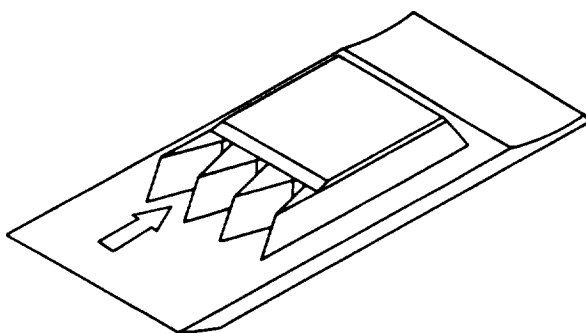


Fig. 4 Geometry for scramjet inlet system.

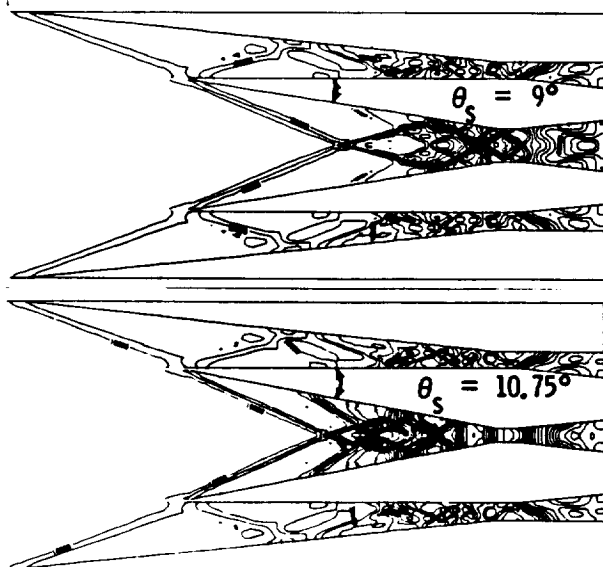


Fig. 5 Pressure contours in a cross plane of a scramjet inlet located at 12% of the inlet height from the cowl plane.



Fig. 6 Surface-pressure map for SSME powerhead.

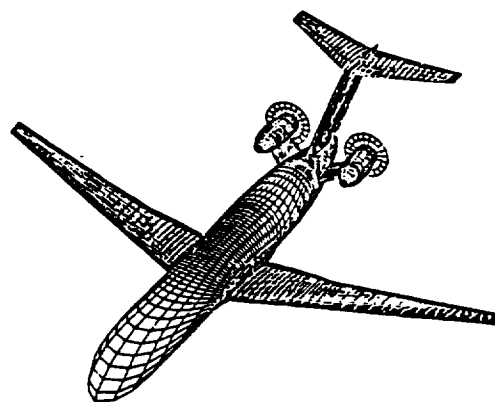


Fig. 7 Surface grid for commercial aircraft configuration with aft-mounted ultrabypass engines.

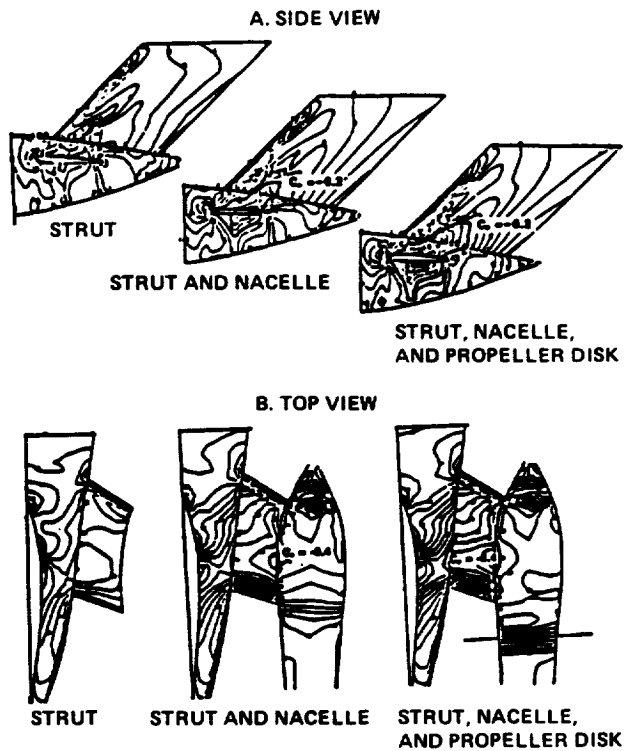


Fig. 8 Isobars from Euler solution in vicinity of ultrabypass engine.

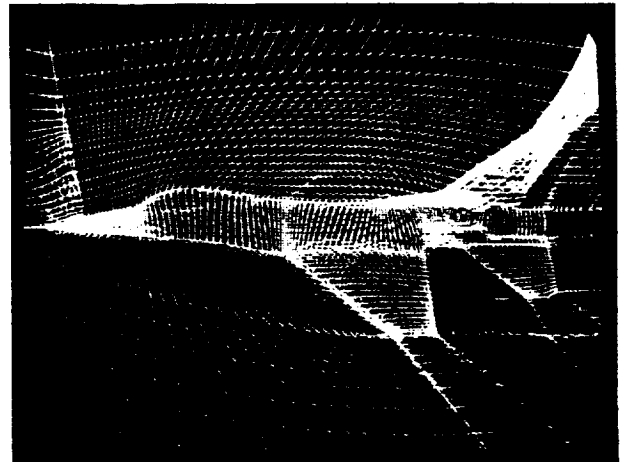


Fig. 10 Surface grid for Euler analysis of F-16 fighter aircraft.

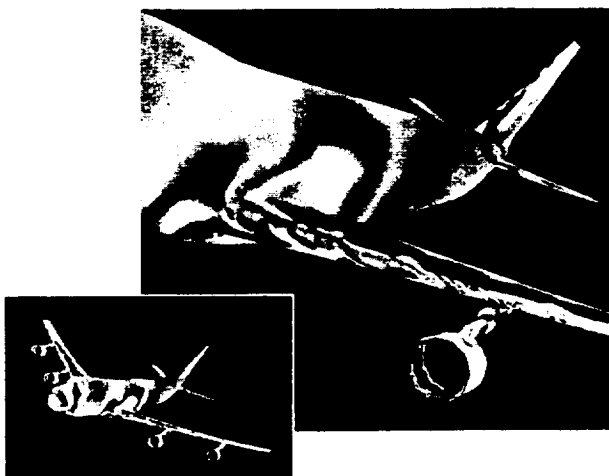


Fig. 9 Pressure from Euler solution contours for B-747-200 commercial aircraft.

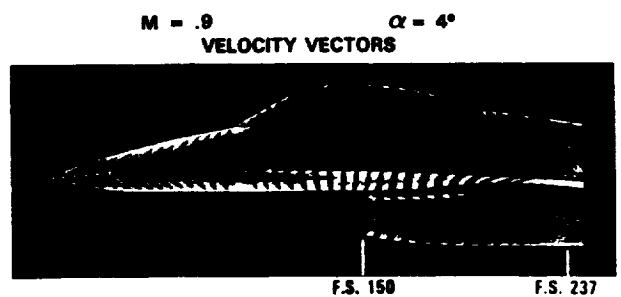


Fig. 11 Velocity vectors for forward fuselage of F-16 aircraft.

MULTIZONE COMPUTATION
 $M_{\infty} = 1.8, \alpha = 0^\circ$

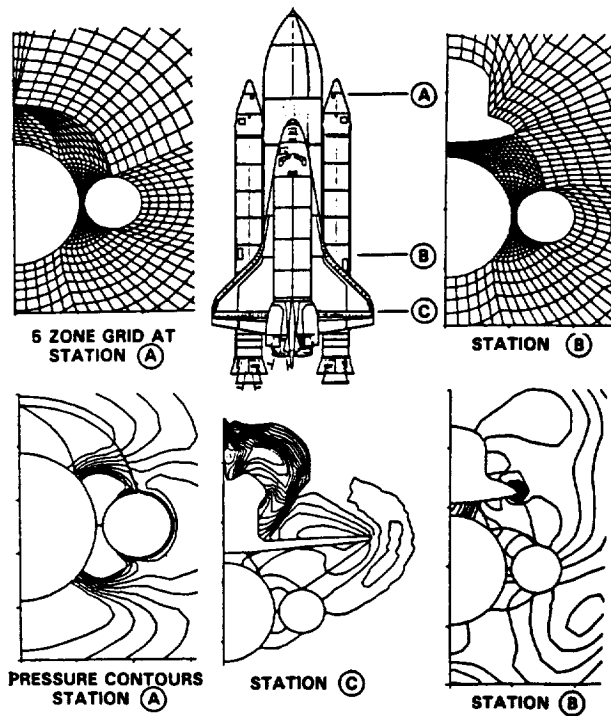


Fig. 12 Geometry for Shuttle, solid rocket boosters, and external tank along with pressure contours from Euler analysis about this configuration.

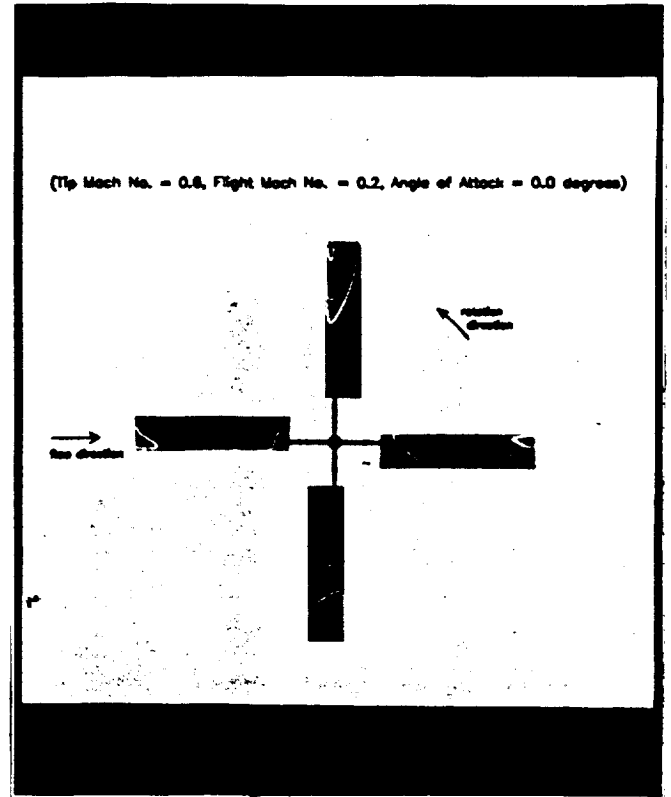


Fig. 14 Surface pressures contours for four-bladed rotor in forward flight.



Fig. 13 F-5 wingtip missile simulation.

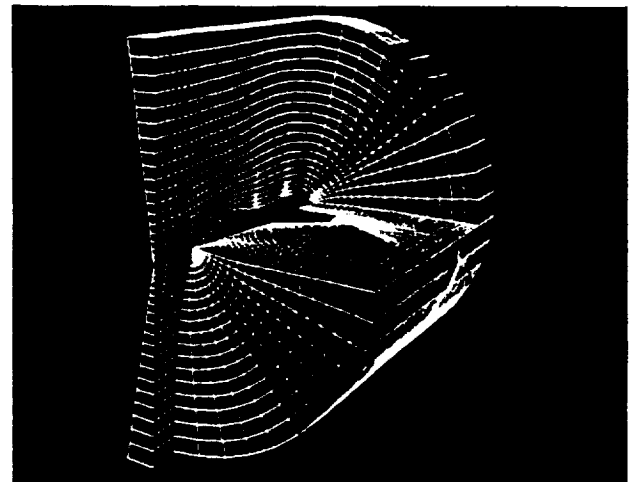


Fig. 15 Grid system for the modified F-16A simulation.

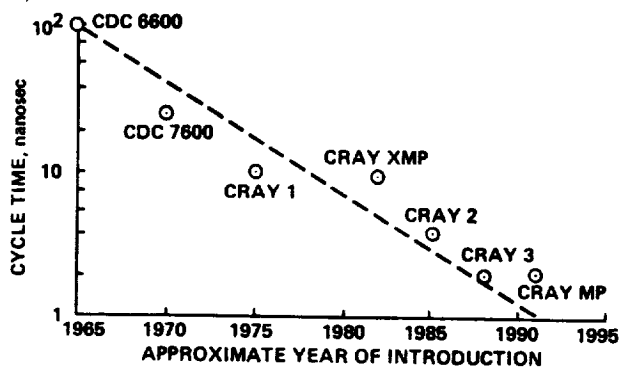


Fig. 27 Decrease in supercomputer cycle time.

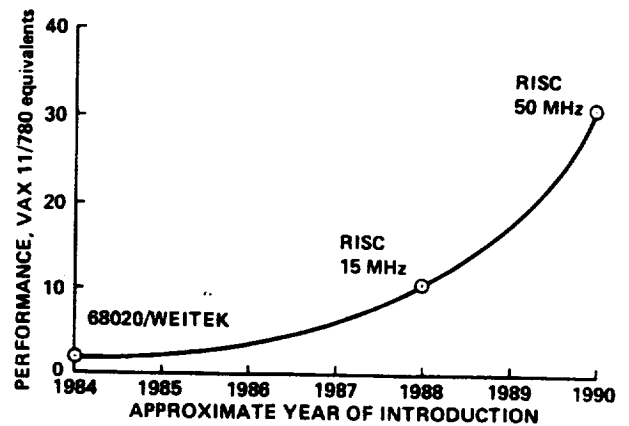


Fig. 29 Increase in work-station performance compared with a VAX 11/780.

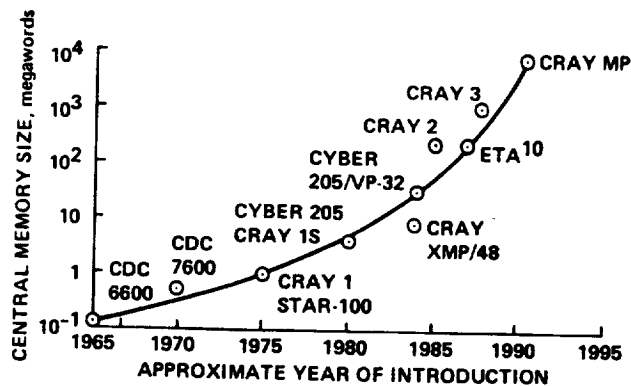


Fig. 28 Growth in supercomputer central memory capacity.

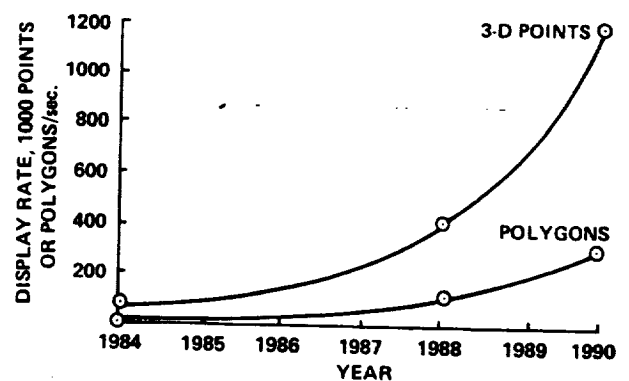


Fig. 30 Projected increases in work-station transformation and display rates.